

ARTICLE

NUMERICAL SIMULATION OF VELOCITY DISTRIBUTION IN THE RIVER LATERAL INTAKE USING THE SSIIM₂ SOFTWARE

Mohammad Reza Borna¹, Mehrdad Farrokhi², Adel Asnaashari³, Omid Alipour Tomaj⁴, Saeed Keramat⁵, Yousef Kor^{6*}

¹PhD Candidate in civil Engineering, Science and Research Branch, Islamic Azad University, Tehran, MSc of Golestan Province Gas Co, Gorgan, IRAN

² Associated Professor, Department of Health in Disaster and Emergencies, University of Social Welfare and Rehabilitation, Tehran, IRAN

³PhD Candidate in Civil-Hydraulic Structure Engineering, Razi University, Kermanshah, IRAN

⁴PhD Candidate, Department of Agriculture and Natural Resources, Gorgan Branch, Islamic Azad University, Gorgan-IRAN

⁵MSc of Hydraulic civils, Water and Wastewater Co of Golestan Province, Gorgan, IRAN

^{6*}MSc of Environmental Engineering, Environmental Health Research Center, Golestan University of Medical Sciences, Gorgan, IRAN

ABSTRACT

In this study, flow numerical simulation has been performed in the direct path of rectangular channel using SSIIM₂ software, that solves Navier-Stokes equations by Finite-Volume Method (FVM). The flow calculations were performed in the three dimensional model using K-ε-RNG and K-ε-Standard turbulence models. In the second section, using this turbulence model, the flow velocity profiles at different sections of the main channel and intake were compared with the experimental and numerical results of other researchers; and a good agreement has been found between them. The comparison of the obtained results with experimental and numerical results of other researchers indicates that this numerical model can well predict the flow velocity profile in the different sections of the main channel and intake.

INTRODUCTION

From the distant past so far, diverting water from its original path has been done for many purposes such as agriculture, urban water supply and etc. Flow diverted into the intake has complex properties and leads to the flow separation zones in the main channel and the intake. That flow enters into the intake, has strong momentum in the direction of main channel and therefore, flow separation occurs through the intake [1]. In this area of the flow, fluid particles in the vicinity of the left wall of the intake channel have rotation mode and in fact, this area of the lateral channel will have no effect on the amount of discharge. In other words, the flow separation zone reduces the discharge of the intake and increase the velocity in other parts of the flow. Also, due to the changes which occur in the velocity distribution in the intake, deposition usually happens in the intake inlet that leads to the reduction in intake efficiency, the entrance of coarse-grained sediments into the network and the increase in executive costs for descaling operations. Any action that reduces the secondary flows and whirlpools at the intake inlet, reduces the accumulation of sediments at the intake inlet and also the sediments enters into the intake. Many studies had been done on diverting the flow in the direct path by previous researchers. Shafaeii Bajestan et al did the experimental and numerical investigation of flow pattern in the intake with 30-degree diversion angle from trapezoidal channel. In this study, firstly, different tests were performed on lateral intake with 30-degree diversion angle from the wall of trapezoidal channel in the laboratory flume and 3D components were measured. Then, this data was used to calibrate and validate the mathematical model SSIIM₂. The results showed that there is a direct relationship between the width of separated flow on the bed and the surface and discharge. Also, they indicated that in compared to discharge from direct channel, the width of flow pipe increases on the surface and decreases on the bed in the discharge from trapezoidal channel and since the concentration of sediment on the bed is more than one in surface layers, obliquity of the wall of main channel will lead to the reduction of input sediment. [2]

Mohammadion et al examined the effect of the curvature of the bottom edge of the 90-degree intersection of direct and arc-shaped channels on the flow pattern. In this research, modification of the flow pattern and reduction of erosion and deposition were examined by replacing the bottom vertical edge of intersection with an arc. The results showed that using an arc with mentioned radius will have significant effect on the reduction of the deposition potential of flow and erosion. [3] Seyyedian and Shafaeii Bajestan determined the dimensions of flow tube and a spiral vortex power in the lateral intakes. In this study, firstly, different tests were performed on lateral intake with 90-degree diversion angle in the laboratory flume and 3D components of velocity were measured. Then, this data was used to calibrate and validate the mathematical model SSIIM₂ and this model was implemented for other hydraulic conditions so that the acceptable range of the data was obtained. The results indicated that the width of flow tube on the bed and the surface is directly related to discharge. Also, the power of secondary flow created in the beginning of the intake was calculated that affects the transfer of bed load to intake and it was found that there is an

KEY WORDS

Ateral intake, flow velocity profile, shear stress, SSIIM₂ numerical model

Published: 10 September 2016

*Corresponding Author

Email: ykor54@yahoo.com
Tel.: +98-9111787338

inverse relationship between it and Froude number and there is a direct relationship between it and discharge. [4] Shakibaeinia et al discussed the application of 3D numerical modeling in simulating the complex phenomena in river engineering by using model SSIIM₂. This study aimed to evaluate the efficiency of three-dimensional numerical models in solving the issues raised in river engineering and validate them in some complex cases. The results of numerical predictions were compared with available experimental or field data. The results indicated the accuracy of mentioned numerical models that can be used in solving the many problems raised in river engineering.[5] .Karami Moghaddam et al. evaluated the effect of the radius of curvature of the intersection zone on the dimensions of separation zone by using 3D model SSIIM₂ and the results were verified by using physical methods.[6] Montaseri et al. [7], in their study, using the flow pattern of the front of the intake, showed that in a 180-degree arc, the flow tube on the front of the intake, which is more in upper layers of flow than lower layers, reduces. Also, the maximum secondary flow power is in the 45-degree cross-section of arc and there is a relative maximum power in the 130-degree cross-section. Law and Reynolds conducted an analytical and experimental study on main and diverted channels with the same width and provided a relationship between the discharge ratio and different Froude number for before and after intersection and the width ratio of two channels. [8,13]. Chen and Lian simulated the t-shaped intersection, which has been studied experimentally by Pop and Salet in two-dimensional form with high Reynolds number by the use of K-ε-Standard model. [9]

The results obtained for small discharge ratio were consistent with the previous experimental measurements but for the larger discharge ratio, their results were fundamentally different from the previous measurements. Issa and oliveira conducted 3D simulation of turbulent flow for T-shaped geometries.[11] Kasthuri and Pundarikanthan performed an experimental study on measuring the length and width of flow separation zone in the intake with 90-degree diversion angle from a rectangular channel and concluded that the dimensions of flow separation zone depend on discharge ratio and the width and length of vortex zone decrease at the inlet of intake channel by increasing the discharge ratio. [12]

MATERIALS AND METHODS

In order to simulate the deposition and investigate the intakes and rivers hydraulically the SSIIM₂ software was designed. This software solves the Navier-Stokes equations by the use of finite-volume method. Finite-volume method is based on direct discretization of the integral form of conservation laws in physical space. Flow analysis is done in steady state and the algorithm SIMPLE is used for coupling the velocity and the pressure. Discretization method of the momentum equations, drop and turbulence kinetic energy and Reynolds stress is the second order forward difference method and discretization method of the equation of pressure is a standard method. Turbulence models used in this software are K-ε-Standard turbulence model, K-ε-RNG turbulence model, local K-ε turbulence model based on the water velocity, K-ω turbulence model with Wilcox's wall law and K-ω turbulence model with k-epsilon wall law. Given differential form of

$$\text{conservation law } \frac{\partial U}{\partial t} + \bar{\nabla} \cdot \bar{F} = Q,$$

RESULTS

the important step in finite volume method is integrating governing equations on control volume:

$$\int_{\Omega_J} \frac{\partial U}{\partial t} d\Omega + \int_{\Omega_J} \bar{\nabla} \cdot \bar{F} d\Omega = \int_{\Omega_J} Q d\Omega$$

$$\int_{\Omega_J} \bar{\nabla} \cdot \bar{F} d\Omega = \int_S \bar{F} \cdot d\bar{S}$$

Using Gauss divergence theory, :

Integral form of conservation law for each control volume, Ω_J , related to the point J is as following:

$$\frac{\partial}{\partial t} \int_{\Omega_J} U d\Omega + \int_{S_J} \bar{F} \cdot d\bar{S} = \int_{\Omega_J} Q d\Omega$$

The above equation is replaced by its discrete form where the integral of volume is expressed as the averaged values of the cell and the integral of surface is expressed as a total of volume package.

$$\frac{\partial}{\partial t} (U_J \Omega_J) + \sum_{faces} \bar{F} \cdot \Delta \bar{S} = Q_J \Omega_J$$

Governing equations of the movement of fluid are the equation of continuity and momentum that are as equations (5) and (6) for turbulent flow in incompressible fluid in a 3D geometry, respectively. In different turbulence models, the turbulence kinetic energy is defined as equation (7). (15)

$$\frac{\partial \bar{U}_i}{\partial x_i} = 0 \quad K = \frac{1}{2} \overline{U_i U_j} \quad \frac{\partial \bar{U}_i}{\partial t} + (\overline{U_j}) \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + g_{xi} + \frac{\partial}{\partial x_j} [v \frac{\partial \bar{U}_i}{\partial x_j} - \overline{U'_i U'_j}]$$

Where $\rho \bar{u}_i \bar{u}_j$ is Reynolds stress, U_i and U_j are the flow velocities in the directions of x and y , t is time, ν is a molecular viscosity, p is pressure, k is turbulence kinetic energy, ρ is fluid density and g_{xi} is gravity in the direction of x_i .

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_T \partial k}{\sigma_k \partial x_j} \right) + p_k - \varepsilon \quad \nu_T = c_\mu \frac{K}{\varepsilon^2} P_k = \nu_T \frac{\partial U_j}{\partial x_i} \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right)$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_T \partial \varepsilon}{\sigma_\varepsilon \partial x_j} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} P_k + C_{\varepsilon 2} \frac{\varepsilon^2}{k}$$

P_k is defined as follows,

K is marked as ε and becomes as follows,

In above equation, P_k is a term of producing turbulence and empirical constant values used in this study are as follows (15)

$$C_\mu = 0.09, \quad C_{1\varepsilon} = 1.43, \quad C_{2\varepsilon} = 1.92, \quad \sigma_\varepsilon = 1.3, \quad \sigma_k = 1$$

Experimental model performed by Barkdoll et al.

In the experimental study by Barkdoll et al. (8), the lengths of main and secondary channels are 2.74 m and 1.68 m, respectively and the diversion angle is considered 90 degrees. The cross sections of both channels are rectangle. Input discharge of main channel (Q_1) is 0.011 m³/s, the depth of flow (d) is 0.31 m and the Reynolds number (Re) is 49600 and the width of both channels (b) is 0.152 m. [Fig. 1] shows a schematic design of the channel. The hydraulic characteristics of water flow are listed in [Table1]

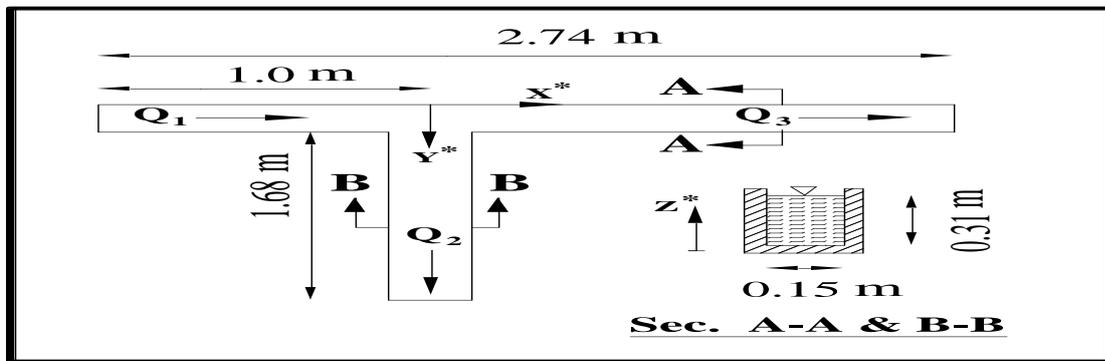


Fig.1: Geometric characteristics of laboratory flume.

Table1. Hydraulic characteristics of flow

d (m)	Q_1 (Lit/s)	R	Q_3 (Lit/s)	Q_2 (Lit/s)	Re	Fr
0.31	11	0.31	7.59	3.41	49600	0.13

Networking and boundary conditions of computing field to simulate experiments done by Barkdoll et al. In this study, the inlet of main channel uses the boundary condition of specified velocity with the average velocity equal to 0.236 m/s. according to the value considered in experimental model, intake discharge ratio (R) is equal to 0.31. Given the inconsiderable changes at the water surface, the symmetry boundary condition is applied to the surface. Wall boundary conditions are applied to rigid boundaries and the walls are considered smooth hydraulically. Also, one of the important factor affecting the speed of the implementation of the model is an appropriate regional networking in which there is flow. Figure-2 shows the plan and perspective of the networking of computing field in 90-degree intake. Table2 lists the number and dimensions of its different zones in the directions of x , y and z .

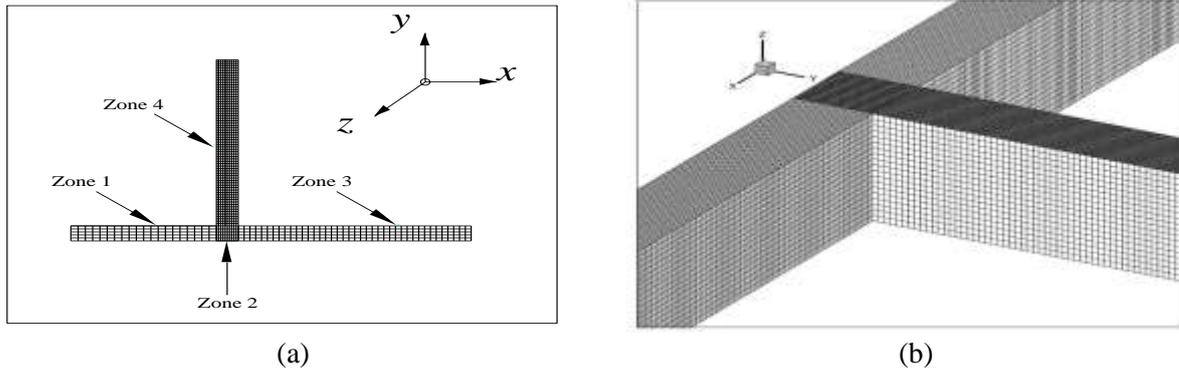
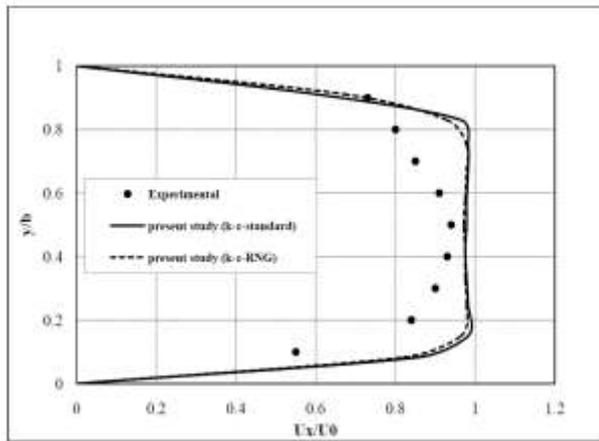


Fig. 2: Networking of computing field within the intake in two modes (a) and (b) perspective.

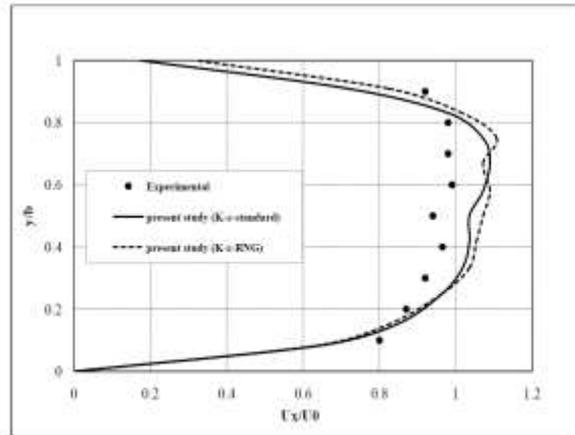
Table 2. The number and dimensions of the zones of computing field in different directions

Zone	number of cells in direction of x	number of cells in direction of y	number of cells in direction of z	Dimensions of cells in direction of x (mm)	Dimensions of cells in direction of y (mm)	Dimensions of cells in direction of z (mm)
Zone1	25	120	19	7.70	6	16.31
Zone2	25	50	19	6	3	16.31
Zone3	25	240	19	6.93	6	16.31
Zone4	25	50	19	3	11.20	16.31

According to experimental study, indicates the dimensionless speed profiles (U_x/U_0) near the water surface for the different cross-sections of main channel with constant input discharge of $0.011\text{m}^3/\text{s}$, intake discharge ratio (R) of 0.31 and Froude number of input flow (Fr) of 0.13. Y^* and X^* are the distances in the axes of y and x that made dimensionless with the width of intake channel. U_x is a maximum velocity at the cross-section $X^*=-4.65$ which is equal to 0.28 m/s. [Fig. 3]. Profiles of computing velocity in different cross sections of main channel (a) $X^*=-4.65$, (b) $X^*=-0.5$, (c) $X^*=0.5$ and (d) $X^*=2$



(a)



(b)

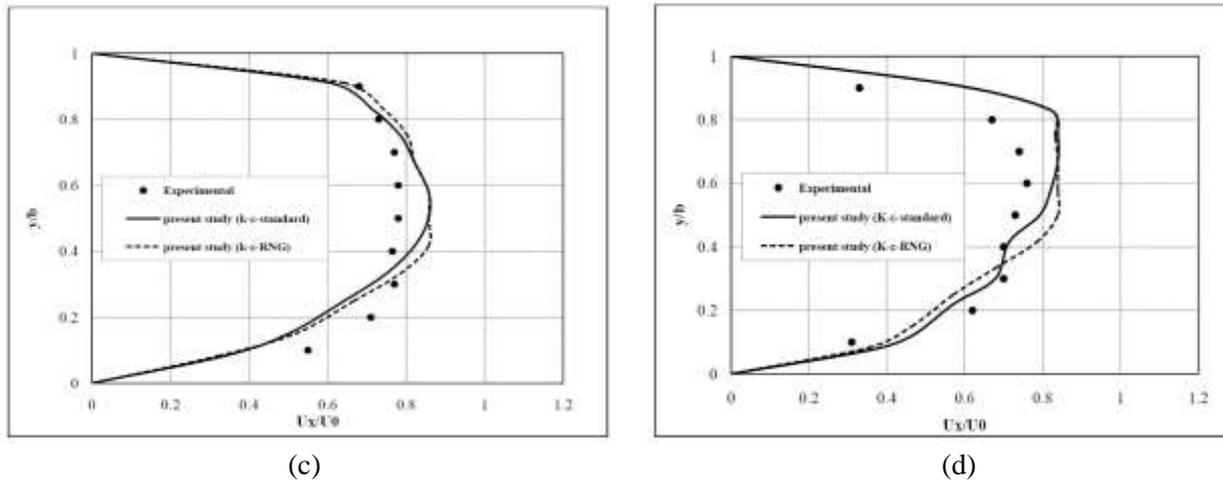
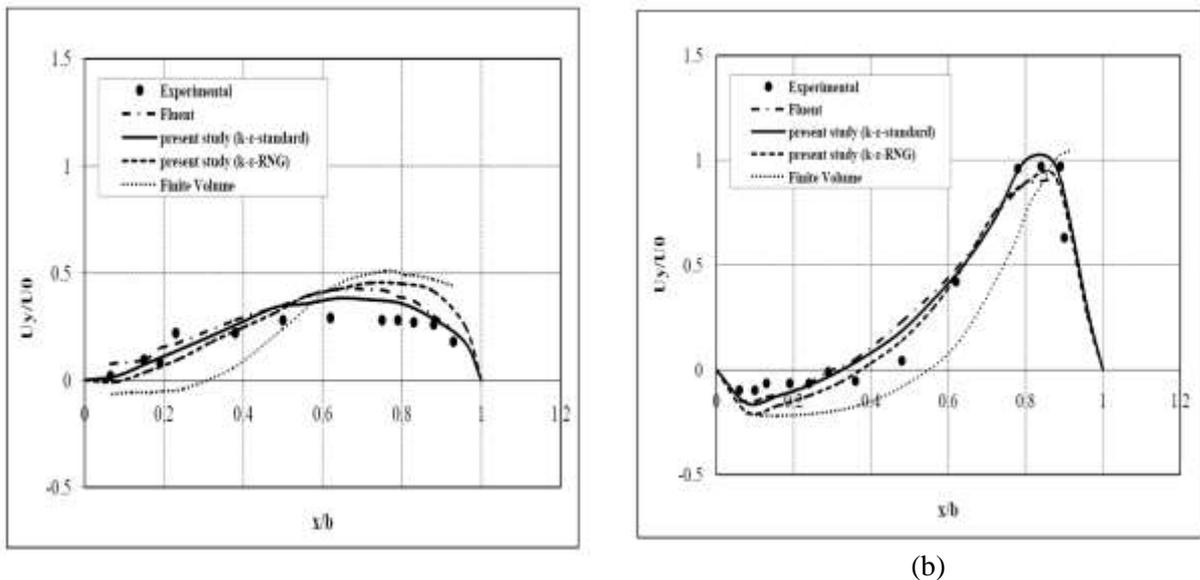


Fig. 3: Examination of flow velocity profiles.

Given to [Fig.(3-a)], in the distances before the inlet of the intake, the profile of velocity keeps its expanded mode and by approaching the inlet, due to the sucking flow applied by the intake, the profiles of velocity are diverted towards the diverted channel and the maximum velocity is moved towards the intake inlet ($X^*=-0.5$) [Fig. 3-b].

The results of this study showed that when the flow enters into the intake, the velocity decreased along the intake inlet and in the bottom wall of inlet ($X^*=-0.5$), the maximum speed receded from the inner wall of the main channel. Residual flow expands along the main channel after passing the intake inlet but, due to the effects caused by the curvature of flow lines in the intake inlet, the maximum velocity deviates towards the inner wall again. Amount of predicted velocity is greater than the amount of experimental velocity, because air tension is neglected in numerical modeling. Figure 4 shows the profile of dimensionless velocity (U_y/U_0) near the water surface for different cross sections. The numerical results obtained from the Fluent software are derived from the numerical study by Safarzadeh and Salehi Neishabori (1) in which K- ϵ -Standard turbulence model was used to examine the distribution of the flow velocity profiles. Also, the numerical results of Finite-Volume are derived from numerical study by Godarzizadeh et al.in which K- ϵ -RNG turbulence model was used(10).

[Fig. 4]. Profiles of computing velocity in different cross sections of the intake (a) $Y^*=1$, (b) $Y^*=2.5$, (c) $Y^*=4$ and (d) $Y^*=10$ According to [Fig. 3 and 4], K- ϵ -Standard turbulence model is better than K- ϵ -RNG turbulence model for estimating the velocity and the results show that the positive and negative amounts of velocity obtained by it are consistent with the results of experimental study. Amount of predicted velocity is greater than the amount of experimental velocity, because the effects caused by secondary flow leads to the transfer of maximum velocity to the bottom of the water surface. [Table 3] shows the average percentage error obtained from comparing the numerical amounts of the software of SSIIM₂ and Fluent with experimental amounts for different sections of main channel and intake.



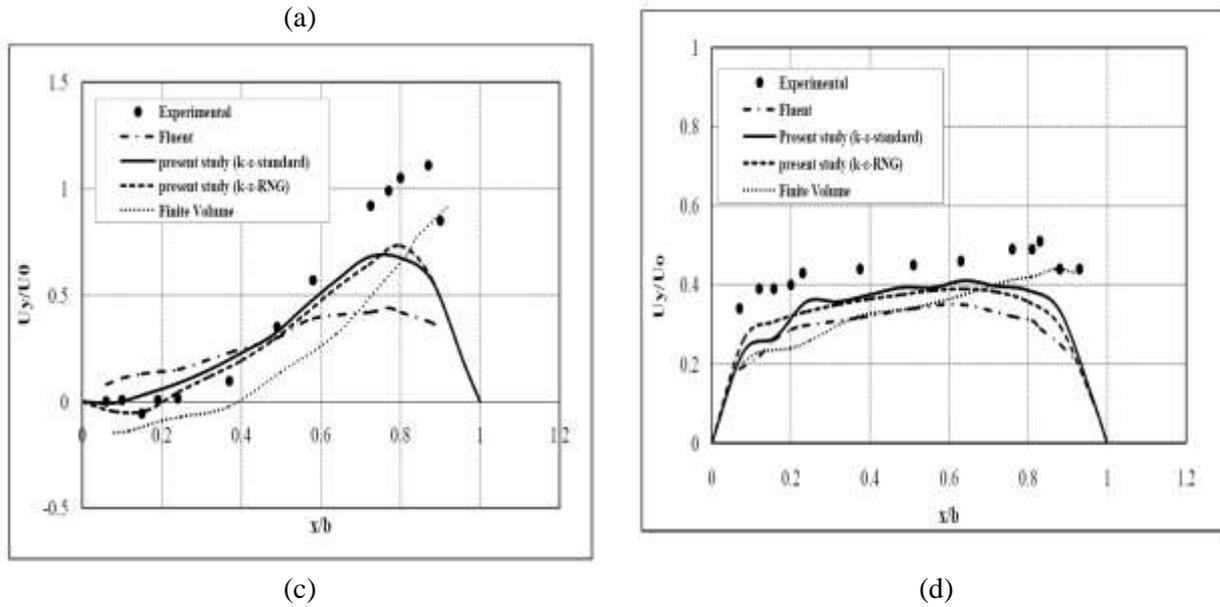


Table3: Average of percentage error obtained from comparing the numerical results of velocity profile in different cross sections of two channels with experimental results

Section	X*				Y*			
	-4.65	-0.5	1	2	1	2.5	4	10
SSIIM2 (K-ε-Standard)	13.28	9.67	12.30	10.71	5.73	15.25	20.60	9.14
SSIIM2 (K-ε-RNG)	13.59	10.72	13.47	11.42	11.20	20.76	22.02	10.05
Fluent					6.41	11.98	41.33	14.67

In the second part of the study, according to the good results obtained from the first part, the profiles of velocities averaged in the depth in different sections of main channel and intake of experimental model provided by Shettar and Murthy examined by K-ε-standard turbulence model.[17]

CONCLUSION

Totally, the results obtained for the flows with applied conditions are as follows:

In the distances before the intake inlet, the profile of velocity keeps its expanded mode and by approaching the inlet, due to the sucking flow applied by the intake, the profiles of velocity are diverted towards the diverted channel and the maximum velocity is moved towards the intake inlet ($X^*=-0.5$) [Figure 3b]. When the flow enters into the intake, the velocity decreased along the intake inlet and in the bottom wall of inlet ($X^*=-0.5$), the maximum speed receded from the inner wall of the main channel. Residual flow expands along the main channel after passing the intake inlet but, due to the effects caused by the curvature of flow lines in the intake inlet, the maximum velocity deviates towards the inner wall again.

K-ε-Standard turbulence model is better than K-ε-RNG turbulence model for estimating the velocity and the results show that the positive and negative amounts of velocity obtained by it are consistent with the results of experimental study.

In both modellings, given the percentage error obtained from comparing the numerical amounts with experimental amounts, the numerical amounts obtained from the numerical model of SSIIM₂ software in most cross sections are more consistent with experimental results than the numerical results obtained from the numerical model of Fluent software, it shows the high ability of this numerical model in determining the velocity profile in different cross sections of main channel and intake.

The maximum shear stress occurs in the right wall of the beginning of intake channel and its value is equal to 2.6 N. Since the erosion and corrosion of walls increase with the increase in shear stress, it can be concluded that the probability of the erosion and corrosion in this zone is more than other zones.

CONFLICT OF INTEREST
None

ACKNOWLEDGEMENTS

None

FINANCIAL DISCLOSURE

None

REFERENCES

- [1] Safarzadeh and Salehi Neishabori [1] Numerical modeling of the three dimensional flow pattern of the lateral intake", First National Congress of Civil Engineering, Sharif University of Technology.
- [2] Shafaeii Bajestan et al, [2] Experimental and Numerical study of flow pattern in the intake with a diversion angle of 30-degree in trapezoidal cross section", Journal of Science and Technology of Agriculture and Natural Resources, Water and Soil Sciences, 15TH Year, Issue 57, Iran
- [3] Mohammadion S, Salehi Neishabori .A, Parhizkar, H, Wahhabi, H, [2013] "The effect of curvature of the bottom edge of the intersection of 90 degrees of direct and arc-shaped channels on the flow pattern", National Conference on Civil engineering and Sustainable Development with a Focus on Reducing Risks in Natural Disasters", Khavaran Higher Education Institution, Mashhad, Iran
- [4] Seyyedian M, Shafaeii Bajestan M, [2011] Determination of the dimensions and power of the spiral vortex power in the lateral intakes", Journal of Water and Sewage, Number 4, Isfahan, Iran
- [5] Shakibaei Nia, A, Zerati, A.R, Majdzadeh Tabatabai, M.R, [2008] "Application of three-dimensional numerical simulation of complex phenomena in river engineering", Journal of The College of Engineering, Volume 42, Issue 4, Pp. 443-455, Tehran, Iran
- [6] Karami Moghaddam M, Seyyedian SM, Shafaeii Bajestan, M. [2008] Examination of flow pattern in the intake with the diversion angle of 90-degree with curvature in inlet by the use of model SSIM2 and comparing it with physical model", 4th Congress of Civil Engineering, Tehran University, Iran
- [7] Montaseri H, Ghodsian M, Shafaiefar M, Salehi Neishabouri, A.A, Dehgahni AA. [2008]. "Experimental investigation of 3D flow field and scouring in a 180 degree bend with a lateral intake. Gorgan J of Agri. Sci. and Natural Resources, 15 [2], 225-533.
- [8] Barkdoll BD, Hagen BL, Odgaard AJ.[1998] Experimental Comparison of Dividing Open Channel with Duct Flow. Journal of Hydraulic Engineering, 124(1): 92-95.
- [9] Chen H B, Lian GS. [1992]. The Numerical Computation of Turbulent Flow in T-junction. Journal of Hydrodynamics, No.3, pp. 50-58.
- [10] Goudarzizadeh R, Hedayat N, Mousavi Jahromi SH.[2010] Three dimensional Simulation of Flow Pattern at the Lateral Intake in Straight Path Using Finite Volume Method. World Academy of Science, Engineering and Technology, pp. 656-661.
- [11] Issa RI, Oliveira PJ.[1994] Numerical Prediction of Phase Separation in Two-Phase Flow Through T-Junctions. Computers and Fluids, 23(2): 347-372.
- [12] Kasthuri, B, and Pundarikanthan, N.V.1987. Discussion on Separation Zone at Open Channel Junction. Journal of Hydraulic Engineering, 113(4): 543-548.
- [13] Law, S. W, and Reynolds, A. J.1966. Dividing flow in an Open Channel. Journal of Hydraulic Engineering. 92(2): 207-231
- [14] Neary, V.S, Sotiropoulos, F, and Odgaard, A. J.1999. Three dimensional Numerical Model of Lateral Intake inflows. Journal of Hydraulic Engineering, Vol. 125, No. 2, pp. 126-140.
- [15] Olsen N B R.[2006] A Three dimensional Numerical Model for Simulation of Sediment Movements in Water Intakes with Multiblock Option. Department of Hydraulic and Environmental Engineering, The Norwegian University of Science and Technology.
- [16] Ramamurthy AS, Junying Qu, Diep Vo.[2007] Numerical and Experimental Study of Dividing Open-Channels Flows. Journal of Hydraulic , 133(10):1135-1144.
- [17] Shettar AS, Murthy K K.[1996] A Numerical Study of Division Flow in Open Channels. Journal of Hydraulic Research, 34(5): 651-675.